REPORT DOCUMENTATION PAGE

Form Approved
OMB NO. 0704-0188

1. AGENCY USE ONLY *(Leave blank)*	2. REPORT DATE	3. REPORT TYPE AND DATES COVERED
	01/08/99	Final Progress: 9/93-9/98

4. TITLE AND SUBTITLE

Finite Element Computation of the Dynamics of Large Ram Air Parachutes

5. FUNDING NUMBERS

DAAH04-93-G-0514

6. AUTHOR(S)

William L. Garrard, Tayfun Tezduyar

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Department of Aerospace Engineering and Mechanics
University of Minnesota
107 Akerman Hall
Minneapolis, MN 55455

8. PERFORMING ORGANIZATION REPORT NUMBER

I97-1

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)

U.S. Army Research Office
P.O. Box 12211
Research Triangle Park, NC 27709-2211

10. SPONSORING / MONITORING AGENCY REPORT NUMBER

ARO 32487.19-MA

11. SUPPLEMENTARY NOTES

The views, opinions and/or findings contained in this report are those of the author(s) and should not be construed as an official Department of the Army position, policy or decision, unless so designated by other documentation.

12 a. DISTRIBUTION / AVAILABILITY STATEMENT

Approved for public release; distribution unlimited.

12 b. DISTRIBUTION CODE

13. ABSTRACT *(Maximum 200 words)*

This research is aimed at simulation of the aerodynamics/dynamics of large parafoils. Parallel computation methods for 3D simulation of the dynamics and fluid dynamics of a parafoil with prescribed, time-dependent shape changes were studied. The mathematical model was based on the time-dependent, 3D Navier-Stokes equations governing the incompressible flow around the parafoil and Newton's law of motion governing the dynamics of the parafoil, with the aerodynamic forces acting on the parafoil calculated from the flow field. The computational methods developed for these 3D simulations include a stabilized space-time finite element formulation to accommodate for the shape changes, special mesh generation and mesh moving strategies developed for this purpose, iterative solution techniques for the large, coupled nonlinear equation systems involved, and parallel implementation of all these methods on scalable computing systems.

14. SUBJECT TERMS	15. NUMBER OF PAGES
High Performance Comuputing, Parafoils	3
	16. PRICE CODE

17. SECURITY CLASSIFICATION OR REPORT	18. SECURITY CLASSIFICATION OF THIS PAGE	19. SECURITY CLASSIFICATION OF THIS ABSTRACT	20. LIMITATION OF ABSTRACT
UNCLASSIFIED	UNCLASSIFIED	UNCLASSIFIED	UL

FINITE ELEMENT COMPUTATION OF THE DYNAMICS OF LARGE RAM AIR PARACHUTES

FINAL PROGRESS REPORT

WILLIAM GARRARD
TAYFUN TEZDUYAR

1/8/99

U. S. ARMY RESEARCH OFFICE

GRANT NUMBER

DAAH 04-93-G-0514

DEPARTMENT OF AEROSPACE ENGINEERING AND MECHANICS
ARMY HIGH PERFORMANCE COMPUTING RESEARCH CENTER
UNIVERSITY OF MINNESOTA
MINNEAPOLIS MN 55455

FINITE ELEMENT COMPUTATION OF THE DYNAMICS OF LARGE RAM AIR PARACHUTES

A. STATEMENT OF PROBLEM STUDIED

Ram air inflated gliding parachutes, called parafoils, are being developed for the recovery of large loads. Using automatic on-board guidance, parafoils have the potential for precise landing control which cannot be approached by circular parachutes. The objective of this project was to develop methods for predicting the aerodynamic performance of large parafoils during inflation, gliding, and maneuvering. These methods allow the accurate prediction of performance during staged inflation, turning, steady glides, and flaring maneuvers. The approach is to use modern, finite-element, three-dimensional CFD methods coupled with massively parallel processing.

The motivation for the project is the lack of a data base and/or scaling laws which are sufficient to provide adequate information for design of large parafoils. This makes it very important to develop analytical methods for predicting the performance of such parachutes. Until recently, the prediction of parachute performance was based primarily on empirical or semi-empirical methods since the solution of the equations governing parachute aerodynamics was beyond the computational capabilities available. Recently CFD methods have been applied to circular parachutes; however, most methods for parachute design still rely on relatively simple mathematical models coupled with data bases from existing parachute systems. Since the accuracy of these methods depends to a large extent on empirical data, their predictive capability for canopies which differ substantially from those used in existing systems is limited.

Simulation of parafoils is an inherently very challenging task in computational fluid dynamics. Various stages involved in the operation of these parafoils, such as the initial stages of inflation, transformation to a fully deployed parafoil, steady gliding, flare maneuver, and landing, makes the task very complex. The ram-air parachutes considered can be as large as 10,800 square-feet (60 ftx180 ft), with a payload capacity of as much as 42,000 lb. The scale of these parachute systems, combined with the various stages of complex behavior described above, makes wind tunnel tests impossible. Therefore, simulation and modeling, with the recent advances in computational methods and computing platforms, becomes a credible, promising complement to costly drop tests.

Previous investigators have limited their mathematical modeling efforts to the steady-state gliding phase of parafoil operations. Standard airfoil theory is modified to attempt to predict lift to drag ratios and other important aerodynamic parameters. Methods range from classical low aspect ratio aerodynamic theory to two-dimensional potential flow methods; however, all of these methods provide overly optimistic estimates of performance. Recently some work using two-dimensional panel methods, two-dimensional Navier-Stokes methods and three-dimensional panel methods has been applied to the analysis of parafoils. There has been no other successful attempt to apply CFD to model the inflation of ram air inflated parachutes.

The approach used in this research is based on finite element solutions of the unsteady, 3D Navier-Stokes equations. The solution of the Navier-Stokes equation is accomplished on massively parallel computers. The advantage of the Navier-Stokes approach is that new configurations for which data is not available can be analyzed from first principles. The disadvantage is that the computations are complex and require a great deal of time even on a very fast computer. These disadvantages can be overcome by using the Navier-Stokes solution to generate lift and drag area data which can then be incorporated in the flight mechanics simulations. The solution of the Navier-Stokes equations can be

regarded as a numerical experiment in which generated data can be used in simple flight mechanics simulations appropriate for design and trade off studies.

The successful use of CFD methods in the aerodynamics of parafoils depends on the availability of large efficient computers. It is only because of the efficient parallel implementation and the computational capability offered by this class of super computers that large-scale, unsteady 3D simulations such as the ones performed are feasible. These computations also require a strong software base and a good understanding of both CFD methods and the physics of parachutes. The benefit is the ability to simulate phenomena which have never been successfully modeled before and to obtain results which should be useful in predicting the performance of large parafoils.

B. SUMMARY OF THE MOST IMPORTANT RESULTS

The basic strategy in our approach to the simulation of large parafoils has been to divide the problem into a number of computationally tractable stages, and use for each stage a mathematical model as realistic as we can handle in terms of the capabilities of our methods and limitations of the computing platform available. Different levels of simplifying assumptions are used. Gradually, the simplifying assumptions are removed. For example, in this project we assumed that the changes in the parafoil shape were prescribed. This assumption relieves us from the additional challenge of solving for the structural deformation of the parachute. However, the methods are currently being extended to solution of the membrane equations assumed to be modeling the structural deformation, and simulation of the parafoil behavior in coupled, fluid-structure interaction. As part of our stage-by-stage approach, we carried out simulations for initial stages of inflation, transformation to a fully deployed parafoil, and steady glide. Later we escalated the level of sophistication to coupled aerodynamics-dynamics interactions, specifically the flare maneuver stage.

Two methods for opening force analysis were developed. In one approach, the lift and drag areas of the parafoil were assumed to vary with time in a pre-specified manner and the point mass flight mechanics equations were solved as a function of time to yield the forces during opening. This method requires empirical data in order to model the lift and drag as a function of time; thus, its predictive value is limited to parachutes which are similar to those for which data is available. The second method is based on finite element solutions of the unsteady, 3D Navier-Stokes equations on massively parallel computers. The advantage of the Navier-Stokes approach is that new configurations for which data is not available can be analyzed from first principles. The disadvantage is that the computations are complex and require a great deal of time even on a very fast computer. These disadvantages can be overcome by using the Navier-Stokes solution to generate lift and drag area data which can then be incorporated in the flight mechanics simulations. The solution of the Navier-Stokes equations can be regarded as a numerical experiment in which generated data can be used in simple flight mechanics simulations appropriate for design and trade off studies.

During the initial phase of inflation, the parachute is modeled as a rectangular box which expands in both the chord-wise and span-wise directions. This box falls under the influence of the suspended payload. The staged opening of the parachute following the initial inflation is modeled by evolving the rectangular box into a curved shape with inflated cells which represents an inflated parafoil. The span of this parafoil increases with time to simulate the sequential opening process. During this stage, the parafoil starts to develop significant lift and begins to glide. The forces and velocities calculated are similar to those measured during flight tests.

The mathematical model used in the computations consists of two components: 1) time-dependent, 3D Navier-Stokes equations governing the incompressible flow around the parafoil; and 2) Newton's law of motion governing the dynamics of the parafoil. The aerodynamic forces acting on the parafoil, which are needed in determining the motion of the parafoil, are derived from the computed flow field. For the aerodynamic computations, the position and displacement rate of the internal boundaries of the computational domain (i.e. the canopy) are derived from the computed motion of the parafoil. These calculations need to be done in a coupled fashion. In our model we assume that the change in the shape of the parafoil is prescribed as a function of time based on shape and size information from design data.

The changes in the shape of the computational domain makes this problem a special case of a more general class of flow problems with moving boundaries and interfaces. The location of the boundaries and interfaces is unknown, and needs to be determined as part of the overall solution. There are a number of alternatives in terms of handling this general class of problems. A stabilized space-time finite element formulation with time-varying spatial domains is the method used. This method, named Deformable-Spatial-Domain/Stabilized Space-Time (DSD/SST) formulation, was developed for solution of flow problems with fluid-object and fluid-structure interactions, moving mechanical components, and free surfaces and two-fluid interfaces. In DSD/SST method, the variational formulation of the problem is written in a space-time domain, and thus automatically takes into account the time-variation of the spatial domain. Linear, or possibly higher-order, interpolation functions, which are continuous in space but discontinuous in time, are used. Because the functions are discontinuous in time, the computations are carried out one space-time slab at a time, without adding a real fourth dimension to the computations. Still, the method is costlier than semi-discrete finite element methods, and should be used, in our opinion, only for complex problems that require a formulation that can handle moving boundaries and interfaces.

As the spatial domain changes with time, the finite element mesh spanning that domain needs to be updated. This is an important component of methods for moving boundaries and interfaces. If not managed effectively, mesh update cost might overwhelm the cost of the computation. For example, if we remesh (i.e. generate a new set of nodes and elements) too often in problems with complex geometries, the cost of automatic mesh generation could become a major part of the overall computational cost. We have developed advanced mesh moving methods which reduce the frequency of remeshing.

For simulation of parafoils with prescribed shape changes, we developed a special algebraic mesh generation and mesh moving method which requires no remeshing throughout the entire simulation. In 3D computation of flows with moving boundaries and interfaces, the finite element discretizations in space and time generate very large coupled, nonlinear equation systems. These equations need to be solved at every time step of the simulation. For this purpose we use the Newton-Raphson method. At each Newton-Raphson step, we need to solve a linear equation system to compute the increment vector. These linear systems are solved iteratively, where at each iteration the residual of the system is formed by an element- vector-based (matrix-free) method. With this, we use a diagonal preconditioner and the GMRES update technique. We have implemented our methods on distributed-memory parallel computing platforms such as the Thinking Machines CM-5, CRAY T3D and T3E, as well as the shared-memory parallel computing platforms such as the multi-processor SGI systems.

Flare maneuver simulation involves solution of over 3.6 million coupled, nonlinear equations at every time step of the computations. These computations were carried on the CM-5. A number of flare maneuver simulations were performed to study the effect of rigging angle on flare performance. The predicted reduction in velocity subsequent to flare was found to be in reasonable agreement with flight mechanics data for typical large

parafoil systems. Sparse-scheme based iterative update techniques have been developed and implemented. These are more efficient than their element-vector based counterparts and can be used for shorter solution time when sufficient memory is available. Currently a fluid-structure interaction capability for studying the canopy deformations in detail is being developed.

C. LIST OF ALL PUBLICATIONS AND REPORTS

Journals

T. Tezduyar, S. Aliabadi, M. Behr and S. Mittal. Massively parallel finite element simulation of compressible and incompressible flows. *Computer Methods in Applied Mechanics and Engineering*, **119**:157-177, 1994.

S. Mittal and T. Tezduyar. Parallel finite element simulation of 3D incompressible flows: fluid-structure interactions. *International Journal for Numerical Methods in Fluids*, **21**:933-953, 1995.

V. Kalro, S. Aliabadi, W. Garrard, E. Tezduyar, S. Mittal and K. Stein. Parallel finite element simulation of large ram-air parachutes. *International Journal for Numerical Methods in Fluids*, **24**:1353-1369, 1997.

T. Tezduyar, S. Aliabadi, M. Behr, A. Johnson, V. Kalro and M. Litke. Flow simulation and high performance computing. *Computational Mechanics*, **18**:397-412, 1996.

T. Tezduyar, V. Kalro and W. Garrard. Parallel computational methods for 3D simulation of a parafoil with prescribed shape changes. *Parallel Computing*, **23**:1349-1363, 1997.

V. Kalro and T. Tezduyar. Parallel iterative computational methods for 3D finite element flow simulations. *Computer Assisted Mechanics and Engineering Sciences,* **5**:173-183, 1998.

V. Kalro and T. Tezduyar. 3D computation of unsteady flow past a sphere with a parallel finite element method. *Computer Methods in Applied Mechanics and Engineering.* **151**:267-276, 1998.

Chapters in Books

T. Tezduyar, S. Aliabadi, M. Behr, A. Johnson, V. Kalro and M. Litke. *High Performance Computing Techniques for Flow Simulations*, pages 363-397, in M. Papadrakakis, editor, *Parallel Solution Methods in Computational Mechanics.* John Wiley & Sons, 1997.

Conference Proceedings: Invited Papers

T. Tezduyar, S. Aliabadi, M. Behr, A. Johnson, V. Kalro and C. Waters. 3D simulation of flow problems with parallel finite element computations on the Cray T3D. In S. N. Atluri, G. Yagawa, and T. A. Cruse, editors, *Computational Mechanics '95, Theory and Applications, Proceedings of the International Conference on Computational Engineering Science, July 30 - August 3, 1995, Hawaii, USA,* Volume 1, pages 482-487, Springer, 1995.

V. Kalro and T. Tezduyar. Parallel sparse matrix computations in finite element flow simulations on distributed memory platforms. *Numerical Methods in Continuum Mechanics, Proceeding of the International Scientific Conference, September 16-18, 1996, High Tatras, Stara Lesna, Slovakia,* pages 1 - 5, 1996.

V. Kalro and T. Tezduyar. A parallel finite element methodology for 3D computation of fluid-structure interactions in airdrop systems. *Proceedings of the Fourth Japan-US Symposium on Finite Element Methods in Large-Scale Computation Fluid Dynamics, April 2-4, 1998.* Nihon University, 1998.

K. Stein, R. Benney, V. Kalro, T. Tezduyar, J. Leonard and M. Accorsi. Parachute fluid-structure interactions: 3D computation. *Proceedings of the Fourth Japan-US Symposium on Finite Element Methods in Large-Scale Computation Fluid Dynamics, April 2-4, 1998.* Nihon University, 1998.

Y. Osawa, V. Kalro and T. Tezduyar. Multi-domain parallel computation of wake flows around secondary objects. *Proceedings of the Fourth Japan-US Symposium on Finite Element Methods in Large-Scale Computation Fluid Dynamics, April 2-4, 1998.* Nihon University, 1998.

T. E. Tezduyar. CFD methods for 3D computation of complex flow problems. *How to apply CFD to practical problems in environment, buildings and structures, International Workshop on "CFD for Wind Climate in Cities", August 24-26, 1998, Hayama, Japan,* pages 73-91, 1998. International Wind Engineering Forum; Japan Association for Wind Engineering.

Y. Osawa, V. Kalro and T. Tezduyar. A multi-domain method for computation of long-wake flows. *AIA '98, Proceedings of the 2nd Ankara International Aerospace Conference, September 9-11, 1998, Ankara, Turkey, Edited by AIA '98 Organizing Committee,* pages 21-28, 1998.

Conference Proceedings: Contributed Papers

S. K. Aliabadi, W. L. Garrard, V. Kalro, S. Mittal and T. E. Tezduyar. Parallel finite element computation of the dynamics of large ram air parachutes. *AIAA paper 95-1581, 13th Aerodynamic Decelerator Systems Technology Conference and Seminar, Clearwater Beach, Florida, May 15-19, 1995,* pages 278-293, 1995.

W. Garrard, T. Tezduyar, S. Aliabadi, V. Kalro, J. Luker and S. Mittal. Inflation analysis of ram air inflated gliding parachutes. *AIAA paper 95-1565, 13th Aerodynamic Decelerator Systems Technology Conference and seminar, Clearwater Beach, Florida, May 15-19, 1995,* pages 186-198, 1995.

K. Stein, S. K. Aliabadi, W. L. Garrard, V. Kalro and T. E. Tezduyar. Parallel finite element computations on the behavior of large ram-air parachutes. *20th Army Science Conference, Science and Technology for Force XXI, June 24-27, 1996, Proceedings, Volume 1,* pages 403-407, 1996.

V. Kalro, W. Garrard and T. Tezduyar. Parallel finite element simulation of the flare maneuver of large ram-air parachutes. *AIAA paper 97-1427, A Collection of Technical Papers, 14th AIAA Aerodynamic Decelerator Systems Technology Conference, San Francisco, CA, June 3-5, 1997,* pages 19-26, 1997.

K. Stein, R. Benney, V. Kalro, A. Johnson and T. Tezduyar. *AIAA paper 97-0505, A Collection of Technical Papers, 14th AIAA Aerodynamic Decelerator Systems Technology Conference, San Francisco, CA, June 3-5, 1997,* pages 277-284, 1997.

T. Tezduyar, V. Kalro and W. Garrard. *AIAA Paper 99-1712, 15th CEAS/AIAA Aerodynamic Technology Conference and Seminar, June 8-11, 1999, Toulouse, France.*

D. LIST OF ALL PARTICIPATING SCIENTIFIC PERSONNEL

William L. Garrard, Professor

Tayfun E. Tezduyar, Professor

Shahrouz Aliabadi, Research Assistant*

Vinay Kalro, Research Assistant/Research Associate*

Chris Waters, Research Assistant*

Sanjay Mittal, Research Associate

Andrew Johnson, Research Associate

*Received Ph.D. while employed on project